NISTIR 6030

THIRTEENTH MEETING OF THE UJNR PANEL ON FIRE RESEARCH AND SAFETY, MARCH 13-20, 1996

VOLUME 2

Kellie Ann Beall, Editor

June 1997
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

U.S. Department of Commerce
William M. Daley, *Secretary*
Technology Administration
Gary R. Buchula, *Acting Under Secretary for Technology*
National Institute of Standards and Technology
Robert E. Hebner, *Acting Director*

PROGRESS REPORT ON FIRE MODELING
NUMERICAL SIMULATION OF VARIABLE DENSITY FLOW WITH HIGH BUOYANCY

Yoshihiko HAYASHI, Building Research Institute, Tsukuba, Japan
Shuzo MURAKAMI, Tokyo University, Tokyo, Japan
Akashi MOCHIDA, Niigata Institute Technology, Niigata, Japan

Abstract

First, I am sorry that it cannot be said that this paper introduces you to the progress report on fire modeling. I will show you the validation of turbulence models applied to the flowfield around a building. In expectation of further developments in fire modeling, now I am entering the main subject.

Flowfields around bluff bodies can be analyzed using various turbulence models, k- ε two equation model (k- ε model) which is the most widely used for engineering applications, Algebraic Second-moment closure Model (ASM) and Large Eddy Simulation (LES).

Next, a flowfield inside a room on fire, which is characterized by variable density distributions with high buoyance, is analyzed with the k- ε model. This flowfield was predicted by the coupled simulation of convective and radiative heat transfer (case 1) and the result was compared with that of the non-coupled simulation (case 2).

1.COMPARISON OF VARIOUS TURBULENCE MODELS APPLIED TO A BLUFF BODY

1-1.INTRODUCTION

The engineering applications of CFD (Computational Fluid Dynamics) have advanced with the development of various turbulence models, e.g. the k- ε two equation model, the Algebraic Second-moment closure Model (ASM), the Differential Second-moment closure Model (DSM) and the Large Eddy Simulation (LES) etc. The high efficiency and accuracy of these models have been clearly confirmed when they are applied to such simple flowfields as the channel or pipe flows. However their performance is not well established for complex turbulent flowfields.

Here, three turbulence models, k- ε model, ASM and LES, are applied to the flowfield around a bluff body, and thus their relative performances are examined.

1-2.NUMERICAL METHODS AND BOUNDARY CONDITIONS

The standard formulation for k- ε model [1, 2] was adopted. The commonly adopted form for ASM was used following the methods of Rodi [3], Gibson and Launder [4] except for the treatment of the wall reflection term [5]. The values of the numerical constants in ASM follow those proposed by Launder, Reece and Rodi [6] and Launder [7]. In LES, the Smagorinsky subgrid model [8, 9] was applied and the value of 0.12 was selected for the Smagorinsky constant.

A staggered grid was adopted. A second-order upwind scheme was applied for the convective terms in the cases of k- ε model and ASM. A second-order centered difference scheme was adopted for the other spatial derivatives. The Adams-Bashforth scheme was used for time marching. Numerical methods and boundary conditions used for the three simulations are made as identical as possible [10].

1-3.COMPARISON OF MEAN VELOCITY VECTOR FIELD

The mean flowfields around a cube given by wind tunnel testing, k- ε model, ASM and LES are compared in terms of mean velocity vectors in Fig.1. The vertical distributions of velocity behind the model are also shown in Fig.2. As shown in Figs.1 and 2, small differences are observed in some areas, e.g., in the separation region on the frontal corner. Although each turbulence model seems to give good results for the mean velocity vector field, some discrepancies are often observed in the results of simulations when we compare the distributions of surface pressure and turbulence statistics .

1-4.DISCREPANCY IN SURFACE PRESSURE DISTRIBUTION

The surface pressure distribution for a cube is compared in Fig.3. The most noticeable position of the difference is the frontal roof corner where the largest negative value appears. The result of k-ε model deviates greatly at this area. The large negative peak at the corner decreases rapidly in a distribution peculiar to k-ε model. We often get such inaccurate pressure distribution when k-ε model is applied to a bluff body. This is caused by the failure to reproduce the small separation here, as is shown in Fig.4, where the area around the frontal corner is magnified. We can observe a small separation in the results of the experiment, ASM and LES, but not in the result of k-ε model. It should be noted again that the conditions of the numerical methods including boundary conditions are made as identical as possible for the simulations of all three turbulence models.

The discrepancy of surface pressure peculiar to k-ε model is very closely related to the turbulence statistics around the frontal corner. As shown in Fig.5, turbulence energy k is overestimated in the case of k-ε model, which gives rise to a large eddy viscosity ν_t ($\propto k^2/\varepsilon$). Hence, the excessive mixing effect produced by this large ν_t eliminates the reverse flow on the roof, as is shown in Fig.4.

1-5.CONCLUSION

The flowfields around a cube predicted by LES, ASM and k-ε model are compared with the experimental data, and the accuracy of each model is examined.

2.COUPLED SIMULATION OF VARIABLE DENSITY FLOW WITH HIGH BUOYANCY

2-1.NUMERICAL METHODS AND BOUNDARY CONDITIONS [12]

The standard formulation for k-ε model was adopted. Since the flowfield is characterized by variable densities, pseudo compressible model [11] was applied. In case of the coupled simulation of convective and radiative heat transfer, convective heat transfer at the wall was calculated by solving heat balance among conduction, convection and radiation. Emissivity at the wall was 0.95.

The main purpose of this study is to see the validity of the numerical methods applied for the flowfield with high temperature, so chemical production during the combustion was not considered. The heater in the fire room was set to be 500℃ higher than the ambient air. The computational domain covered an area of 1.8m×1.2m×1.2m as shown in Fig.6. This domain was discretized into 30×30×30 meshes. Hybrid scheme was applied for convective terms.

The view factor was calculated using Monte Carlo method and the numerical simulation was conducted.

2-2.RESULTS OF SIMULATION

The mean flowfields given by the simulation are compared in terms of mean velocity vectors in Fig.7. Some differences are observed in some areas, e.g., wall A-B and ceiling A-C. In these areas the mean velocity component of the coupled simulation is larger than that of the non-coupled one. Various discrepancies are also observed even in the result of the coupled simulation, e.g., the floor D-B. In this part the condition of wind vector is different from that of the experiment conducted recently. It seems that the treatment of the constant value of convective heat transfer rate should be modified in the near future.

The temperature distributions are examined in Fig.8. In case of the coupled simulation, boundary layer is observed at each wall. This is due to the higher temperature at the wall in comparison with the non-coupled simulation. However, the temperature gradient along the vertical direction is large only at the vicinity area of the wall. This point is different from the experiment in which the gradient area around the upper part is wider. Therefore another model applied for a large space was proposed [13].

2-3.CONCLUSION

The flowfield inside a room on fire was calculated by the convective-radiative coupled simulation. The result was compared with that of the non-coupled one. There is a large difference between them, so it seems to be important to apply the heat balance on the wall to such flowfields with high buoyancy.

REFERENCES

[1] B.E.Launder and D.B.Spalding, Mathematical Models of Turbulence, Academic Press (1972)

[2] B.E.Launder and D.B.Spalding, The Numerical Computation of Turbulent Flows, Comput. Methods Appl. Mech. Eng., 3 (1974) 269-289

[3] W.Rodi, ZAMM, 58 (1976) T219-T221

[4] M.M.Gibson and B.E.Launder, Ground Effects on Pressure Fluctuations in the Atmospheric Boundary Layer, J. Fluid Mech., 86 (1978) 491-511

[5] S.Murakami, A.Mochida and Y.Hayashi, Scrutinizing k-ε EVM and ASM by means of LES and Wind Tunnel for Flowfield around Cube, 8th Symp. on Turbulent Shear Flows (1991)

[6] B.E.Launder, G.J.Reece and W.Rodi, Progress in the Development of a Reynolds Stress Turbulence Closure, J. Fluid Mech., 68 (1975) 537-566

[7] B.E.Launder, University of Manchester, Institute of Science and Technology, Report No.TFD/82/4 (1983)

[8] J.Smagorinsky, General Circulation Experiments with the Primitive Equations, Part 1, Basic Experiments, Mon. Weather Rev., 91 (1963) 99-164

[9] J.W.Deardorff, ANumerical Study of Three-dimensional Turbulent Channel Flow at Large Reynolds Numbers, J. Fluid Mech., 41 (1970) 453-480

[10] S.Murakami, Comparison of Various Turbulence Models Applied to a Bluff Body, J. Wind Eng. Ind. Aero., 46/47 (1993) 21-36

[11] S.Kato et al., Numerical Simulation of Variable Density Flow with High Buoyancy, Part 5, Summaries of Technical Papers of Annual Meeting Architectural Institute of Japan, D (1992) 521-522, written in Japanese

[12] Y.Hayashi et al., Numerical Simulation of Variable Density Flow with High Buoyancy, Part 15, Summaries of Technical Papers of Annual Meeting Architectural Institute of Japan, D (1994) 39-40, written in Japanese

[13] T.Chikamoto et al., New Low-Reynolds Number k-ε Model Applied to Stratified Flowfield, Part 1, 2, Summaries of Technical Papers of Annual Meeting Architectural Institute of Japan, D (1994) 17-20, written in Japanese

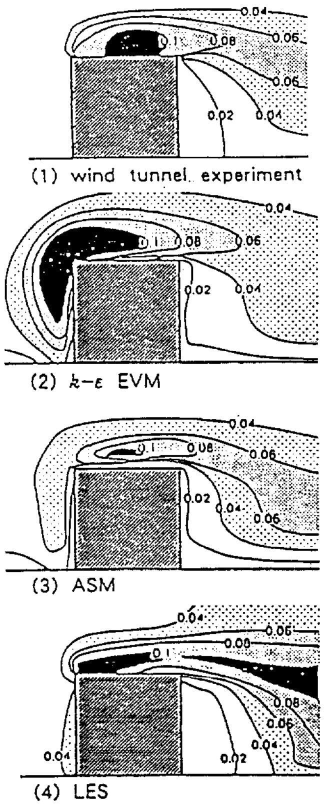

Fig.5 Distri-butions of k at center sec.

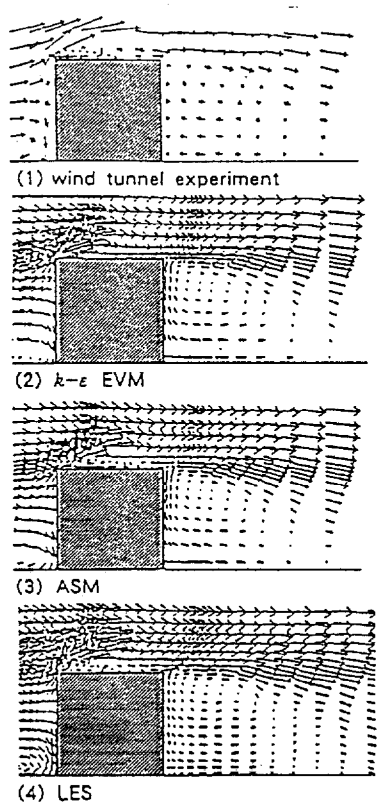

Fig.1 Distri-butions of velocity vecters

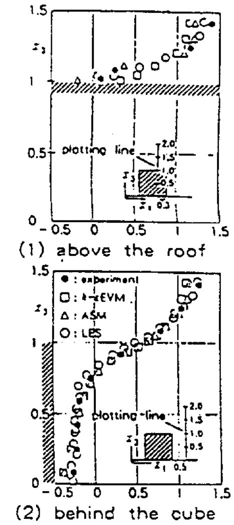

Fig.2 Vertical profiles of $\langle u_1 \rangle$ at center sec.

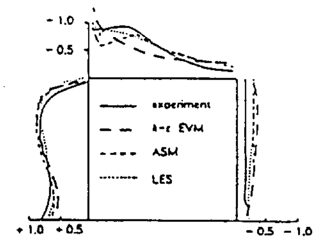

Fig.3 Distri-butions of $\langle C_p \rangle$

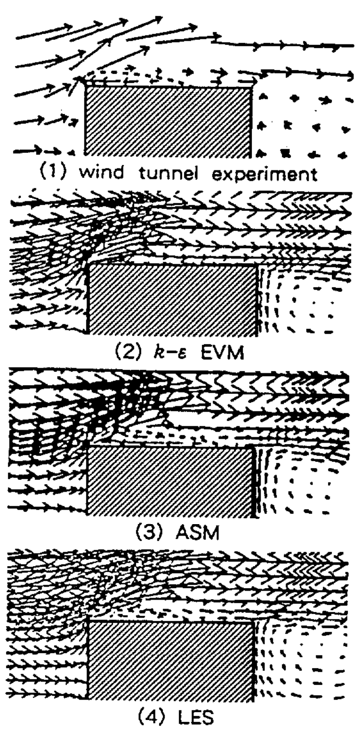

Fig.4 Same with Fig.1 but magnified around frontal corner

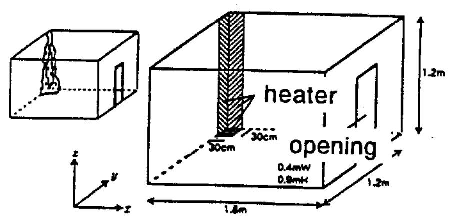

Fig.6-1 Fire room model

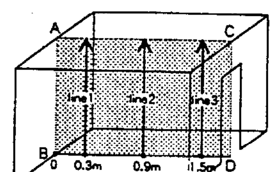

Fig.6-2 Plotting line

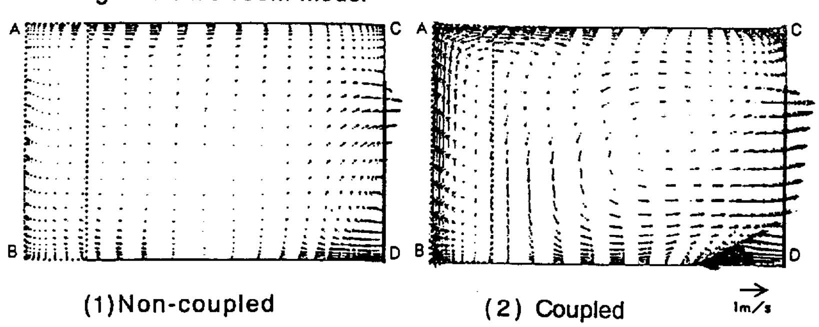

Fig.7 Distributions of velocity vectors

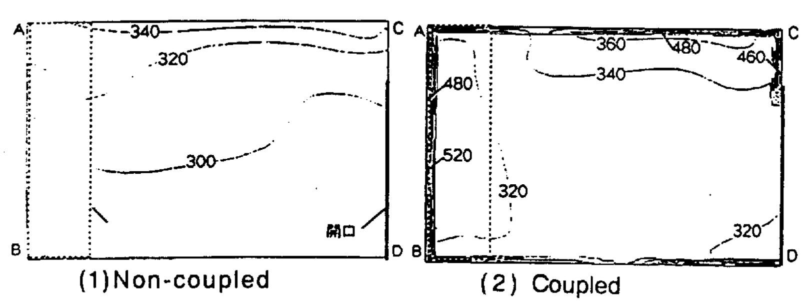

Fig.8 Distributions of temperature